NUMERICAL SIMULATION OF A RESEARCH VESSEL'S AFTPART HULL FORM

Vasilij Djačkov[1], Tomas Žapnickas[1], Jonas Čerka[2], Rima Mickevičienė[2], Žydrūnas Ašmontas[2], Lukas Norkevičius[2], Ieva Ronkaitytė[2], Peilin Zhou[3], Eduardo Blanco-Davis[4]

1. Open Access Centre for Marine Research, Klaipeda University, H. Manto str. 84, LT-92294, Lithuania

2. Faculty of Marine Technology and Natural Sciences, Klaipeda University, Bijunu str.17, LT-91225, Lithuania

3. Department of Naval Architecture & Marine Engineering, University of Strathclyde, 100 Montrose St., G4 0LZ, Scotland UK

4. Department of Maritime and Mechanical Engineering – LOOM Research Institute, Liverpool John Moores University, James Parsons Building, Byrom Street, Liverpool L3 3AF, UK

Abstract: This paper presents results of numerical simulation of the aftpart hull form of a research vessel. The wake rolls onto the transom platform, therefore creating additional resistance to the movement of the vessel. To solve the problem and to propose the optimal transom form, the study was carried out using the method of successive approximations and computational fluid dynamics (CFD) algorithms. The flow lines around the hull of the vessel and the wave pattern behind the hull were analyzed, and the optimal transom form was chosen. Water resistance analysis was carried out to prove the efficiency of the chosen solution.

Keywords: CFD; water resistance reduction; catamaran; hull transom form optimization.

1. Introduction

Research works done in previous century are mainly based on model tests of various vessel forms and calculation methods elaborated in line with such tests [1-5]. However, the calculation methods elaborated on the base of model tests have limitations by the geometrical parameters and shape of hull forms, and in our case cannot be efficiently applied. Last two decades CFD studies are widely used to make research on vessel hull form optimization. By calculation method, they could be categoriesed into studies made using Reynolds-averaged Navier–Stokes equations (RANSE) [6-10] and studies made using boundary element method (BEM) [11, 15]. Dispite many studies have been made in the field of hull form optimization and resistance reduction, the results of these studies are not always used for the creation of hull forms for new vessels. In some cases, it is not always possible to apply the known methods and their results in solving hull design tasks for new ships. In search of a compromise in ship design, some technical and operational characteristics deteriorate in order to implement others, not less important. Therefore, previous experience and expertise may not always be precisely adapted in new vessel designs. This pushes ship-owners to carry out new research, with the aim to find the decision where the hull shape and the elements are optimized. One of the most significant optimization subjects is the stern. The water flow should be smoothly flowing around the stern and separating from it. This is particularly important because a poor design of the stern can cause turbulence and wake, which increases the resistance of the hull. All adverse events usually cannot be avoided, so depending on the type of a ship and the projected needs, the stern is designed in an optimal way for working under certain conditions. The stern is also important in the way that it is usually equipped with propulsion equipment; therefore, its improper design can lead to a negative interaction between the hull and propeller, causing cavitation [12, 13] and noise and vibration [14]. Water flow distribution is also important, because uneven flow to the propeller work area can reduce the effectiveness, thus reducing the speed of the vessel. Optimization of the hull form can reduce hull resistance (essentially wave resistance, by improving the flow around the hull) and, in certain cases, can increase the propulsion thrust. Undoubtedly hull optimization, and the research and tests required, add additional cost and time; but a well-designed vessel will demonstrate higher speed and / or lower fuel consumption, and will result in financial benefits during exploitation. A reliable, well-designed and optimized vessel, will be also environmentally friendly due to the fuel economy and reduction in exhaust emissions.

This paper studies flow formation and distribution over the stern of the Klaipeda University research vessel [16]. The case under the study is a newly built multi-purpose catamaran type research vessel, meant for geological and biological exploration [17]. The length of the vessel is 38.7 meters, and it is equipped with two azimuth thrusters at stern (see Fig. 1).

Fig. 1 Port side and part of the stern of the Klaipeda University Research Vessel.

The vessel reaches its full speed by using only 70% of the total engine power. An increase in power leads to hardly noticeable increase in speed, while vibration also appears in the wheelhouse area. Furthermore, the wake does not separate properly from the transom, and moves together with transom platforms (see Fig.2).

Fig. 2 Wake waters moving together with transom platforms.

Hull form optimization has been chosen as solution to the possible problems rising from the above-mentioned negative effects. Traditional and standard series analysis methods of resistance evolution were not appropriate. Though it is a medium speed vessel with the maximum attainable speed of 12.5 knots (Fn=0.33), the stern structure and its hydrodynamic properties may significantly affect the resistance. The research objective is to investigate and determine the waterflow around the transom stern, and to find the optimal hull form configuration. This study aims not only to optimize the stern form, but also write down an indication for creation of stern body shape for such type of vessels, which can be used to avoid design mistakes in the future.

2. Computational study methodology and approach

Ship flows are described by the Navier–Stokes equations, which for the incompressible flow can be written as follows [1]:

$$\frac{\partial V}{\partial t} = \nu \Delta V - \frac{1}{\rho} \nabla p + F, \qquad (1)$$

where V is the velocity vector field, t – time, ν - kinematic viscosity factor, Δ - vector Laplace operator, ∇ - nabla operator, ρ - density, p - pressure, and F is vector field of mass forces.

One of the weaknesses of CFD is that the results and their accuracy depend on the CFD computer program, the method of calculation, conditions, and other parameters. In order to obtain accurate test data, it is necessary to create the right conditions for the experimental model. The computational mesh parameters were chosen by recommendations for similar study cases and have been described below in the estimation of simulation settings part of this paper. Additionally, a sensitivity analysis should always be carried out, in order to properly validate the results obtained. The reader is advised that one of the shortcomings of the results presented herein, is that a sensitivy and mesh dependency analysis has not been carried out due to some constraints, and it will be presented in the following complex study of this case.

The simulation was carried out using the CFD computer software FLOW-3D. In consideration of the RANS free-surface methods, there are a number of approaches, dealing with the flow conditions at the location of the air–water interface [2]. The approach, used to determine the location of the water free surface, consists in capturing the location implicitly through determining where, within the computational domain, the boundary between air and water is located.

Since FLOW-3D approximation accuracy depends mainly on the number of the mesh elements, to create a precise hull form in CFD software will require an input of a particularly dense mesh. In this case, it will result in a significant increase of calculation time and necessary computer resources. Increasing the quantity of mesh cells until a certain threshold can make the computational calculation become too long or even not possible; therefore, an extremely high number of mesh cells is impractical and requires a lot of calculation time. While approximating the hull form, a problem to replicate the sharp edges of the hull arises.

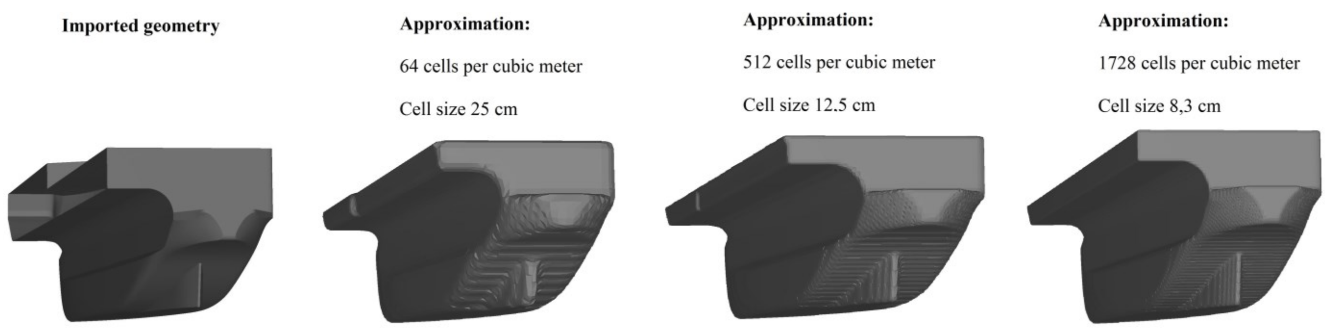


Fig. 3 Imported hull geometry and its approximation in FLOW-3D software.

There is a skeg in the aft of the hull of the vessel that has angular edges, therefore approximating it in CFD software will result in some smoother edges in comparison to reality. Fig. 3 shows the shell approximation with different mesh detailing. By increasing the amount of cell quantity per cubic meter to 4096 (cell size equal 6,25 cm), the computer failed to approximate the hull due to the complexity of calculations. The hull model will be also used as the basis for the creation of alternative forms for flow optimization around the ship hull.

2.1. Estimation of simulation settings

At the first stage, the optimal boundary sizes for the experiment should be estimated while aiming to receive sufficiently accurate results, with a feasible calculation time. The simulation is set with the maximum vessel speed of 12,5 knots (6,43 m/s). Half of the vessel (one hull) is used in the simulation, setting symmetry boundary condition in vessel center line.

The experiments show, that the ratio between the vessel underwater section area and the simulation basin section area should not exceed 1%, and it is recommended to be 0,4% [18, 22]. Under the above recommendation, the transversal section of boundary should be around 52m x 33m [21]. In this case, we have large experiment boundaries, higher number of cells, relatively low detailing of hull form or much longer calculation time with higher cell resolution.

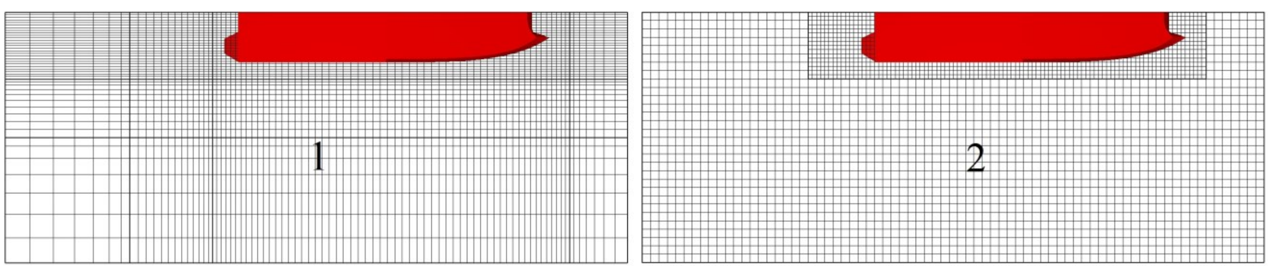
In order to eliminate these disadvantages, it is necessary to use a variable mesh, concentrating more cells where the flow details are most important, e.g. around the vessel hull. The simulation is focused on the aftpart of the vessel MINTIS, where the flow around the hull form (aftpart) is the most interesting, providing initial conditions for the efficient work of the propulsion system.

After some preliminary tests, the simulation specifications were developed. Symmetry boundary condition will be used. The vessel is fixed with the water flowing around it. Mesh block dimensions in length and height will not be smaller than 33 meters. The width of the mesh block must to be also sufficiently big. A variable mesh will be used, concentrating the cell quantity near the vessel hull. The duration of the simulation must be at least 60 seconds.

There are two possibilities for creation of a variable mesh (see Fig. 4):

1. To make the mesh from one block, dividing it into planes, where the size of cells is changing gradually.
2. To make the mesh from blocks with different cell sizes.

When mesh cell sizes are the same, their volumes are also the same; therefore, the energy transition is stable. When variable-sized cells are used in the mesh, then more complicated equations should be used in order to get sufficiently accurate calculation results.



1 - the mesh from one block; 2 - the mesh from blocks with different cell sizes.

Fig. 4 Variable mesh creation possibilities in FLOW-3D software.

While creating a mesh from several blocks, it is necessary to maintain the ratio between cell sizes. The density of the mesh cells is chosen in such a way, that the edges of different meshes would be aligned. This way CFD software can calculate interaction between different elements faster and more accurately. The scheme of the created mesh is shown in Fig. 5. The number of mesh elements is 4 305 070.

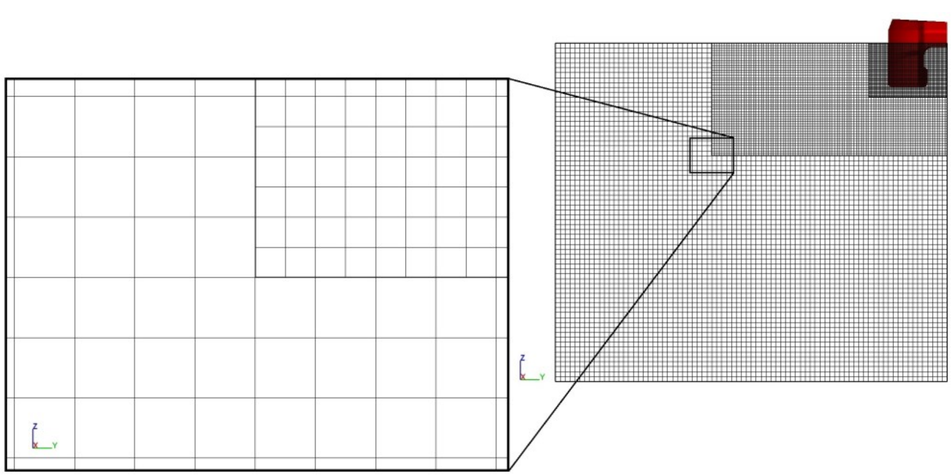


Fig. 5 Mesh created from three blocks with different cell sizes.

The analysis of the experiment conditions showed the main hull hydrodynamic disadvantages, which can be divided into several main groups:

- Decreased flow rate (vortex) at the end of skeg;
- A sudden change of the flow across the lower skeg plane;
- A strong wake, the flow separates disorderly;
- A bow wave partially coincides with a stern wave.

The analysis of flow lines in the aftpart area (see Fig. 6) showed that the flow tends to follow the buttock direction, so that the direction of flow changes, e.g. switches from the horizontal (bottom of skeg and transom area), to the vertical movement following the buttock direction. In this area, the flow suddenly changes its direction becoming turbulent.

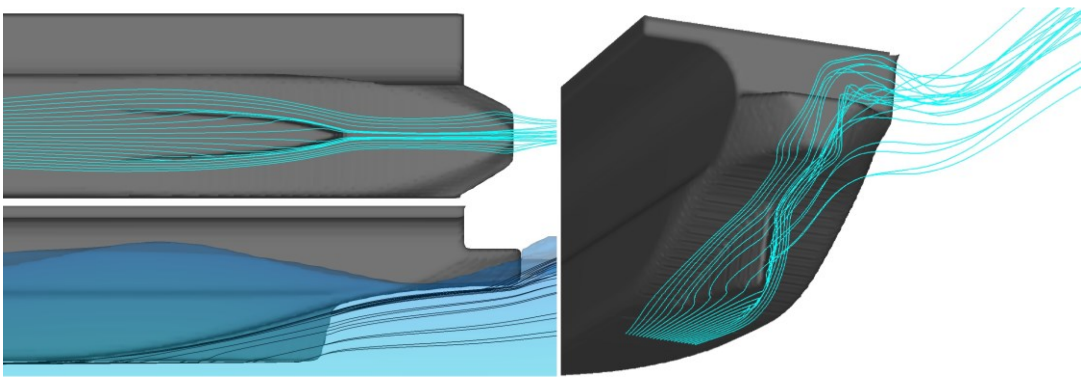


Fig. 6 Waterflow around skeg.

In Fig. 7 it can be noticed that the waterflow speed at the transition point from bottom to sides, following to the transom is increasing significantly, causing appearance of a turbulent flow just at the transom platform of the vessel.

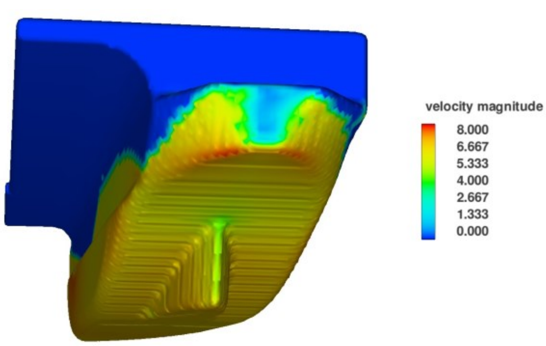


Fig. 7 Distribution of the flow speed over the hull.

In order to implement the research tasks, alternative shape variants will be created and compared. This will be the selection of the best among the analysed variants, which is similar to the direct search method with random potential options. Comparisons of all different hull shape cases are made on equal terms (color scale, flow lines, and imaging parameters of other results being equal). From different research results [2, 19, 20], it is known that a transom should be optimized by changing the form, so that the flow would separate without turbulence. In our case we will use a transom stern form, which ends with sharp edges.

Transom stern is mostly used on high-speed vessels; however, the maximum speed of the vessel MINTIS is not significantly high. Therefore, there is a possibility that the flow will not separate or will separate partly. In this case, the transom stern should be less submersed, or it would be necessary to use other form of aftpart. Another solution to reach the flow separation effect is, when the aftpart wave would damp the forepart wave, or so that at least these waves would not coincide. To fulfil these conditions, it is necessary either to make the vessel longer or to increase its speed. In our case, we cannot change these conditions; therefore, it makes sense that it is the aftpart form the one to be modified.

In view of these considerations, several transom shape modifications were created and are shown in Figure 8. In the case of the first transom variant (Fig. 8, Nr. 1), the created form is massive, which converges very little in the aft direction and is cut at the transom. In the case of second transom variant (Fig. 8, Nr. 2), the principles of transom stern are still followed, but the transom plane is “broken”, for improving the aftpart hydrodinamical qualities by reverse maneuvering.

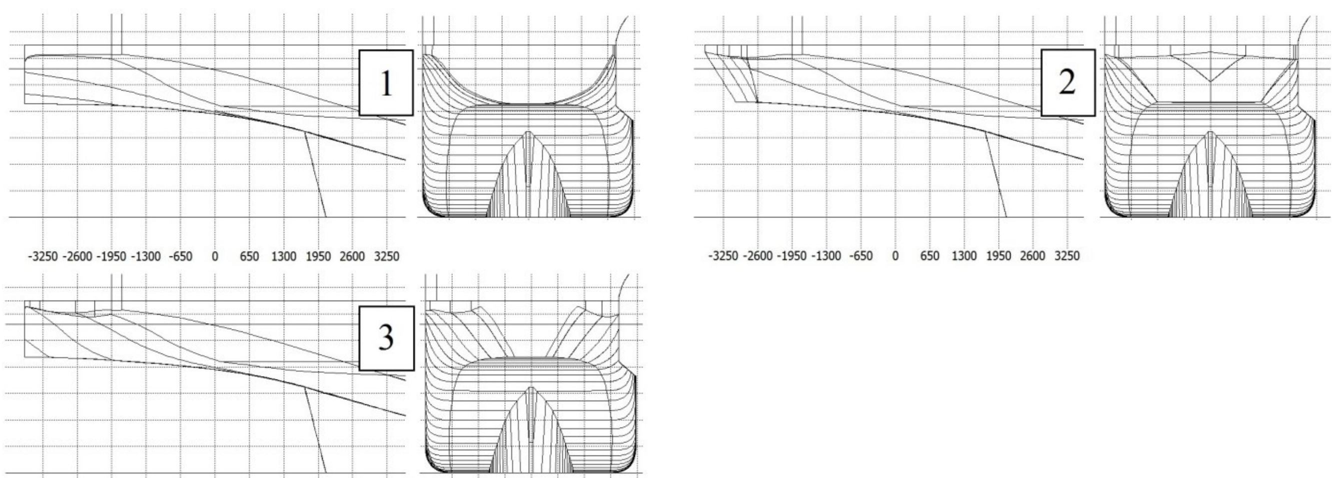


Fig. 8 Transom shape modifications.

The third transom variant (Fig. 8, Nr. 3) has minimal difference with the aftpart of the MINTIS vessel, intending to keep the form of the existing construction; only the rear part is changed by widening it a bit and removing round corners.

3. Research results and analysis

In Fig. 9 the catamaran waves pattern is shown. It can be seen that the forepart wave at the speed of 12.5 knots, coincides with the wave created by the aftpart form. It would be necessary to change main ship parameters by lengthening the aftpart in order to incite the aftpart wave to interfere with the trough of the forepart wave. Depending on the construction of the aftpart, the flow and the free surface are also influenced.

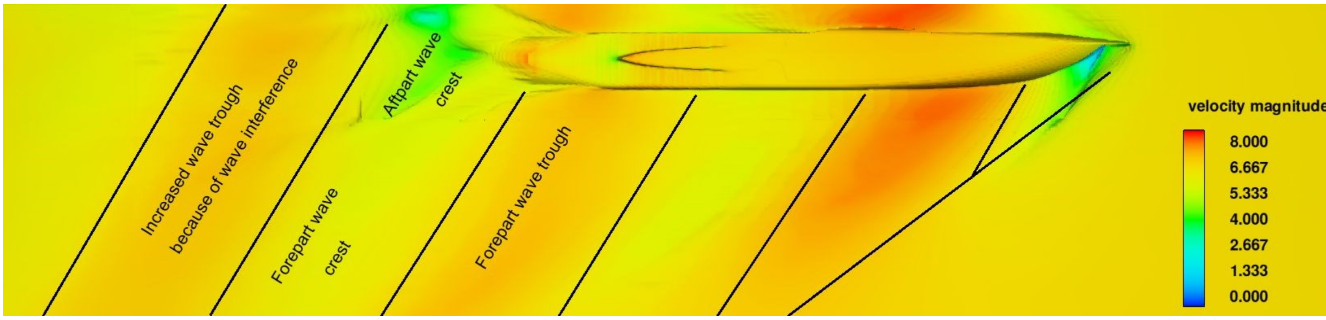


Fig. 9. Transom stern (1 variant) wave interference scheme.

In Fig. 10, the main view on research results is shown. It can be seen that in the "number" 1 variant, the difference is most significant. A comparison of the waves created by the different transom forms is also shown; in Fig. 11, the research results of the vessel MINTIS' aftpart form are compared to results of other alternative aftpart forms. For the transom stern (number 1 variant), the aftpart wave is shifted a bit and therefore interferes with the crest of the forepart wave. In this case, the formed wave and the trough behind it are bigger than in the vessel MINTIS original form. However, the aftpart wave is not coming to the face of the forepart wave anymore, and therefore it is not breaking, and it is not moving together with the transom platform.

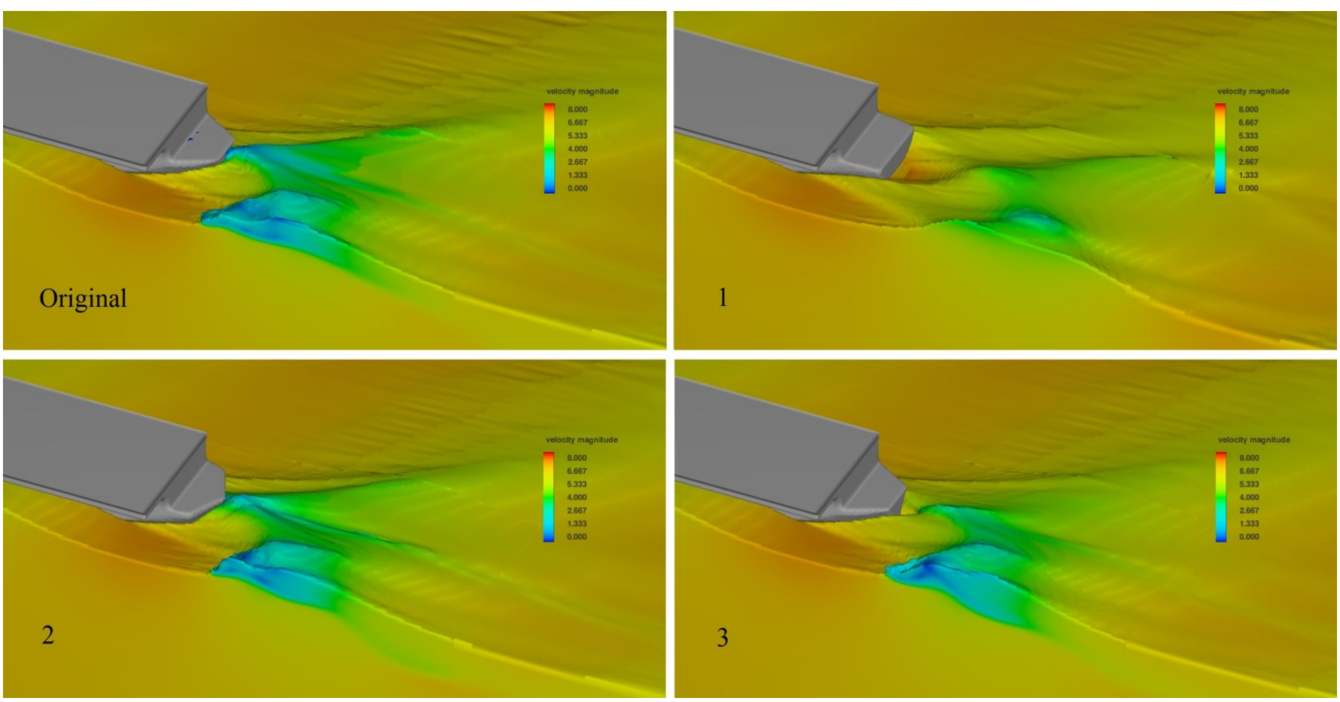


Fig. 10. Distribution of the flow (wave) speed for the aftpart forms under research

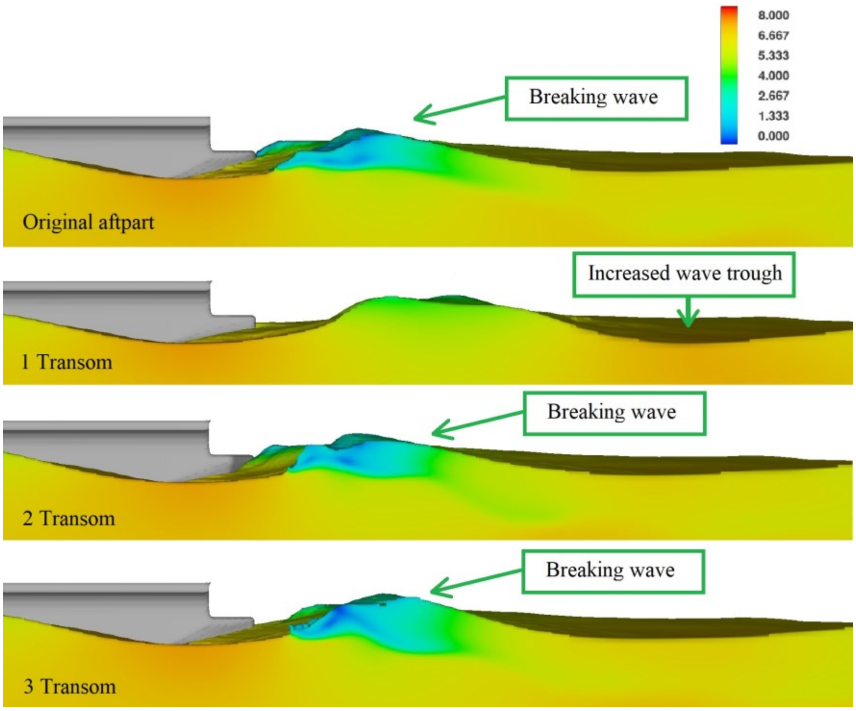


Fig. 11 Flow speed (m/s) in the centerline section of the vessel for different transom form variants.

For the transom variants 2 and 3 the wave profiles are very similar; however, a broken wave is shown a bit more to the aft, in comparison to the original MINTIS aftpart wave system. In the case of the transom variant 3, the flow separates from the aftpart.

The flow configuration and flow rates in the centerline section of the hull are shown in Fig. 12, and the distribution of the flow velocity over the hull for different transom variants is shown in Fig. 13.

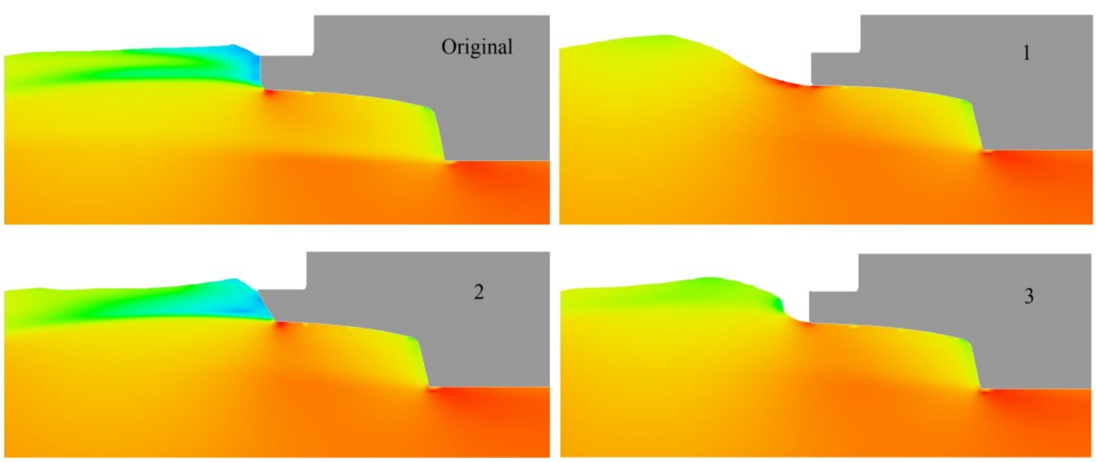


Fig. 12 Flow speed (m/s) in the centerline section of the hull for different transom form variants.

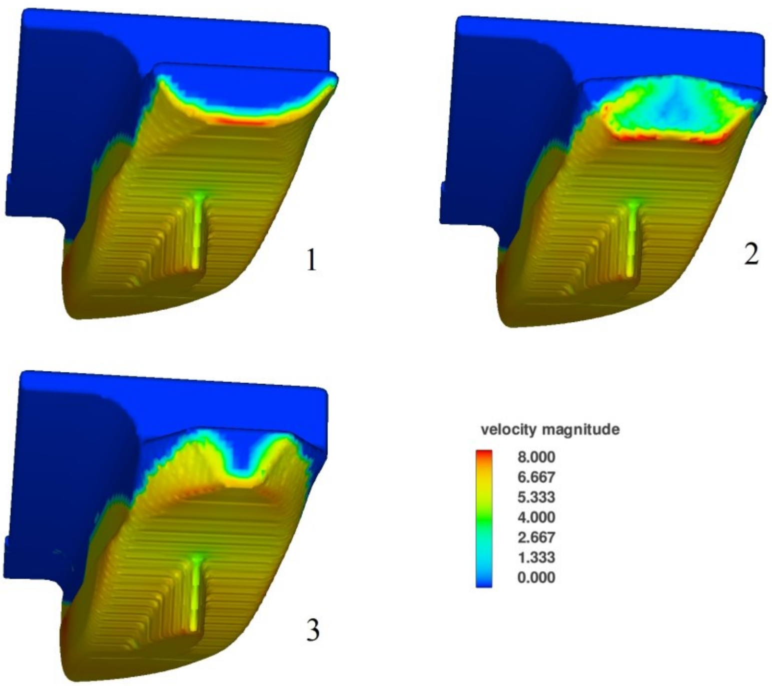


Fig. 13 Distribution of the flow velocity over the hull for different transom variants.

In the case of the transom form 1, the flow is separating from the aftpart and it is not moving together with the transom platform (Fig. 12). In the case of transom form 2 with the broken transom plane, the flow is not separating from the aftpart and therefore is actually moving together with the transom platform (Fig.12). It is very similar to the original transom form, but it has bigger transom plane (Fig. 8) and bigger mass of water moving together with it (Fig.13), what can even result in a higher water resistance. In the case of transom form 3, the flow is separating from the aftpart. In this last case the transom plane is not very big, and the flow passing it around is merging directly. Although the transom is dry, the flow suddenly rises up in velocity. Probably the flow will not separate in reality, especially at smaller speeds, therefore this transom form should be studied in more detail. Nevertheless, this form does not require many changes of the aftpart form; therefore, it would be possible to modify hydrodynamical qualities of the vessel MINTIS, with minimum changes made to the hull. Such modification scheme is shown at Figure 14.

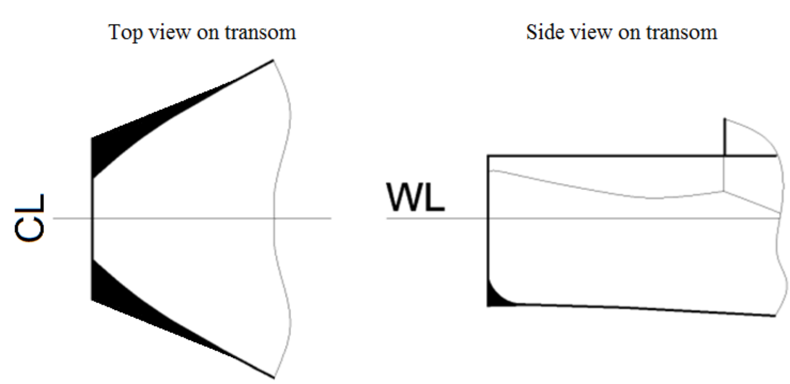


Fig. 14 Suggested modification to the existing aftpart form, seeking to get flow separation at transom area.

To make the forepart wave smaller and the flow separation easier, it would be necessary to lift a bit the aftpart. In such case the flow will separate easier, but it will be more complicated from modification point view, taking in consideration the installation place of the azimuth thrusters.

As it was already mentioned, there is a bigger trough formed behind the aftpart wave. By closer look, it can be seen that two separate waves are forming in the aftpart. One is being formed by the bilge of the vessel and is less visible, while another one is at the end of the vessel form (aftpart). In the case of the vessel MINTIS hull form, both waves are coming to the face of the forepart wave (Fig. 15 Nr. 2); therefore, the broken wave and its reduced flow speed are consequences of these two waves. In the case of the transom stern (Nr. 1 variant), the interference of these waves is observed. The bilge wave travelling in the centerline direction meets the identical wave of the opposite hull (Fig. 15 Nr. 1). The resulting wave is moving away from the centerline, and interferes with the aftpart created wave. At the interference place, moderate raise of the wave and reduced flow speed are observed (Fig. 15 Nr. 3). To reduce the interference of these waves it is necessary to optimize not only the transom form, but to change the complete aftpart form in order to reduce the waves created in the bilge area. However, such serious change of hull is hardly to implement and should studied in the design stage, e.g. before building the vessel.

From the resulting data, a table for comparing transom modification results can be created. This will offer a clear understanding on hydrodinamical and physical changes.

Table 1. Research results for d=2,8m, v_s=6,43m/s.

	Transom form			
	Original	1	2	3
		Change, %		
Displacement, m^3	607,9	0,4	0,0	0,0
Wetted surface, m^2	615,1	0,8	-0,1	0,1
Calm water resistance, kN	114,0	-12,6	+4,9	-3,7

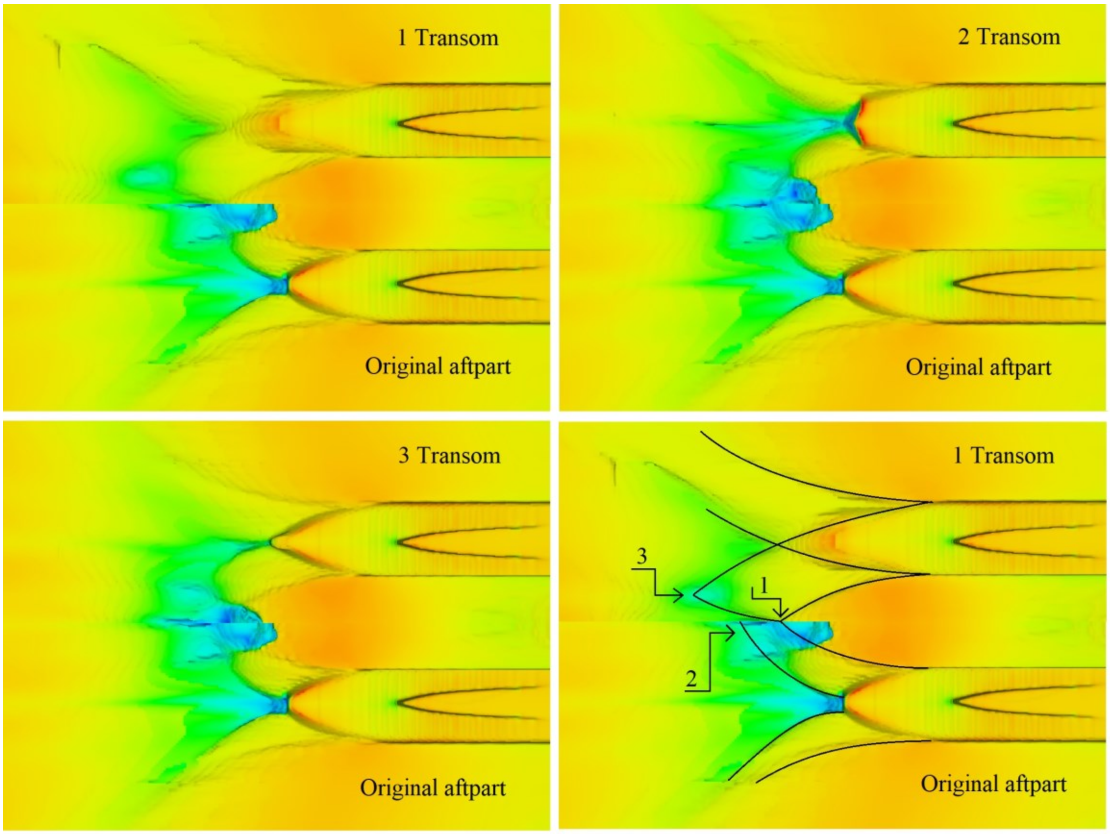


Fig.15 Comparison of aftpart flow speeds (waves)

The data in Table 1 reveals that transom variants 1 and 3, have a lower resistance in comparison to the original one. The decrease in resistance reaches about 12%.

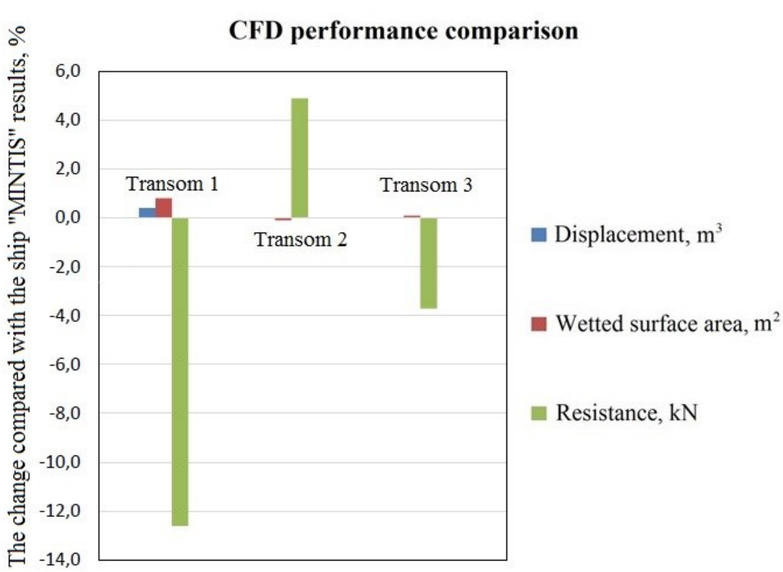


Fig. 16 Comparison of characteristics of alternative transom variants.

The graph in Fig. 16 shows the displacement and wetted surface area of the examined transom forms, in comparison to the original one. Having studied transom modification variants, it can be stated that the number 1 transom variant has the best hydrodinamical qualities and flow separation. Despite the fact that the interference of the forepart and the aftpart waves is quite big and the water displacement and wetted surface area are increased, in comparison to the other modification variants, the water resistance is smaller, because of flow separation. In the case of the third variant of the transom form, the flow separation is also reached, however, the wave in the center line of the hull still remains close to the transom. This variant required the least of the aftpart modification. The second transom form variant is inappropriate for the speed of 12.5 knots, because the desired hydrodinamical qualities are not reached.

4. Conclusions

The CFD simulation of a research vessel showed that at 12.5 knots (6.43 m/s) speed the hull generates a wake, that rolls onto the transom platform. Depending on the data of the investigation, three variants of transom form modifications were developed, considering such vessel modification aspects as minimal changes and practical implementation possibilities, while avoiding complex vessel modifications:

- The first variant is quite different from the original stern form, as its construction affects hydrodynamic qualities. This aftpart configuration gives a good flow separation effect; therefore, the aftpart wave is not running together with the transom. The broken wave in the centerline of the vessel, which appears in other variants, is avoided. The resistance is reduced about 12%.
- The second variant of the aftpart form modification is inappropriate for the speed of 12.5 knots, because the speed is too low for this form for the flow to separate. It also increases the resistance about 5%.
- The third variant of the aftpart form modification reduces the resistance about 4%. Although the resistance reduction is not so obvious as with the transom stern and there is a broken wave created in the centerline of the vessel, the resulting flow is separating from the transom. This form is very similar to the original transom form, and therefore it can be implemented relatively easily.

The results of the research provide preliminary indication for creating the optimal stern body shape during the design stage. During the creation of the bodylines of the vessel, ship designers are mostly trying to find a compromise between the practical arrangement of components of the vessel to find a favorable flow regime. This last principle without implementing the knowledge on flow separation in the aftpart, can have negative effect on hydrodynamic qualities of the vessel. As it was shown in other research works, the transom stern is one of the solutions. In particular study case in order to reach good hydrodinamical qualities, one of the solution is to have the stern optimally submersed, and to remove round corners at the transom plane. However, the wave pattern created by the vessel should also be taken into consideration.

References

1. Voytkunskiy, Y. I., 1988. Ship Theory Handbook. Vol. I. Hydromechanics, Resistance of Ship, Ship propulsion Devices. Leningrad: Sudostroyenie (in Russian).

2. Molland A., Turnock S., Hudson D., 2013. Ship resistance and propulsion: practical estimation of ship propulsive power. Cambridge: Cambridge University Press.

3. Molland, A., ed. 2008. The maritime engineering reference book. Oxford: Butterworth-Heinemann.

4. F. Pérez, J.A. Clemente. Constrained design of simple ship hulls with B-spline surfaces. Computer-Aided Design, Volume 43, Issue 12, December 2011, Pages 1829-1840.

5. Apostolos Papanikolaou. Holistic ship design optimization. Computer-Aided Design, Volume 42, Issue 11, November 2010, Pages 1028-1044.

6. Fuxin HUANG, Chi YANG. Hull form optimization of a cargo ship for reduced drag. Journal of Hydrodynamics, Ser. B, Volume 28, Issue 2, April 2016, Pages 173-183.

7. Chi YANG, Fuxin HUANG. An overview of simulation-based hydrodynamic design of ship hull forms. Journal of Hydrodynamics, Ser. B, Volume 28, Issue 6, December 2016, Pages 947-960.

8. Trong-Nguyen Duy, Takanori Hino, Kazuo Suzuki. Numerical study on stern flow fields of ship hulls with different transom configurations. Ocean Engineering, Volume 129, 1 January 2017, Pages 401-414.

9. Keunjae Kim, Fabian Tillig, Nicolas Bathfield, Hans Liljenberg Hydrodynamic optimization of twin-skeg LNG ships by CFD and model testing. International Journal of Naval Architecture and Ocean Engineering, Volume 6, Issue 2, June 2014, Pages 392-405.

10. Ivo Marinić-Kragić, Damir Vučina, Milan Ćurković. Efficient shape parameterization method for multidisciplinary global optimization and application to integrated ship hull shape optimization workflow. Computer-Aided Design, Volume 80, November 2016, Pages 61-75.

11. Yasser M. Ahmed. Numerical simulation for the free surface flow around a complex ship hull form at different Froude numbers. Alexandria Engineering Journal, Volume 50, Issue 3, September 2011, Pages 229–235.

12. Kwang-Jun Paik, Hyung-Gil Park and Jongsoo Seo. RANS simulation of cavitation and hull pressure fluctuation for marine propeller operating behind-hull condition. International Journal of Naval Architecture and Ocean Engineering, Volume 5, Issue 4, December 2013, Pages 502-512.

13. Artur K. Lidtke, Victor F. Humphrey, Stephen R. Turnock. Feasibility study into a computational approach for marine propeller noise and cavitation modelling. Ocean Engineering, Volume 120, 1 July 2016, Pages 152-159.

14. Batuhan Aktas, Mehmet Atlar, SerkanTurkmen, Weichao Shi, Roderick Sampson, Emin Korkut, Patrick Fitzsimmons. Propeller cavitation noise investigations of a research vessel using medium size cavitation tunnel tests and full-scale trials. Ocean

Engineering, Volume 120, 1 July 2016, Pages 122-135.

15. Vernengo, G., Brizzolara, S. Numerical investigation on the hydrodynamic performance of fast SWATHs with optimum canted struts arrangements. Applied Ocean Research, Volume 63, February 2017, Pages 76-89.

16. Open Access centre for marine research [online], available at: <http://apc.ku.lt/en/index.php/fleet-and-field-research-laboratory/> [Accessed 17 July 2017].

17. Association “Baltic valley” [online], available at: <http://balticvalley.lt/en/wp-content/uploads/2014/11/brosiura_web.pdf> [Accessed 17 July 2017].

18. Flow Science [online], available at: <http://www.flow3d.com/home/resources/cfd-101> [Accessed 23 May 2015].

19. Jing TangXing, ZheTian, XinPingYan. The dynamics of ship propulsion unit – large hull – water interactions. Ocean Engineering, Volume 124, 15 September 2016, Pages 349-362.

20. M. Haase et al., Application of RANSE-based Simulations for Resistance Prediction of Medium-Speed Catamarans at Different Scales. 18th Australasian Fluid Mechanics Conference Launceston, Australia, 3-7 December 2012 [online], available at: http://people.eng.unimelb.edu.au/imarusic/proceedings/18/270%20-%20Haase.pdf [Accessed 20 July 2017].

21. ITTC – Recommended Procedures and Guidelines. Rev. 06, 2014.

22. Mohsin A.R. Irkal, S. Nallayarasu, S.K. Bhattacharyya. CFD approach to roll damping of ship with bilge keel with experimental validation. Applied Ocean Research, Volume 55, February 2016, Pages 1-17.